

# LTSpice tutorial

## Objectives:

In this tutorial, you will learn how to create a hierarchical schematic of an inverter consisting of MOS transistor, resistor and capacitor, using LTSPICE schematic capture tool. You will also learn how to run the simulation and plot the results

## Software

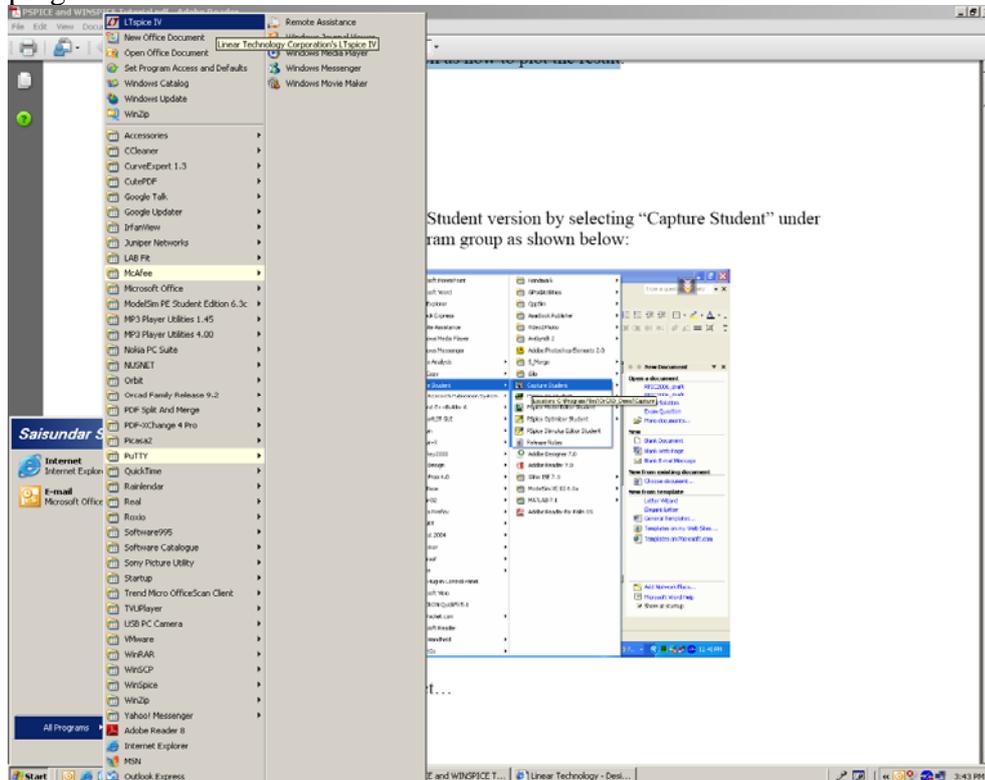
LTSpice IV

## Installation

1. Go to <http://www.linear.com/designtools/software/> and download LTSpice IV.
2. Install it in you computer and in the installed directory do the following
3. copy the **CMOS035.lib** file to which normally would be in a path “C:\Program Files\LTC\LTSpiceIV\lib\sub”
4. copy the nmos\_035.asy and pmos\_035.asy file to which normally would be in a path “C:\Program Files\LTC\LTSpiceIV\lib\sym”

## Tutorial for LT SPICE

- 1) Launch LTSpice by clicking the LTSpiceIV icon on the desktop or from the programs menu.

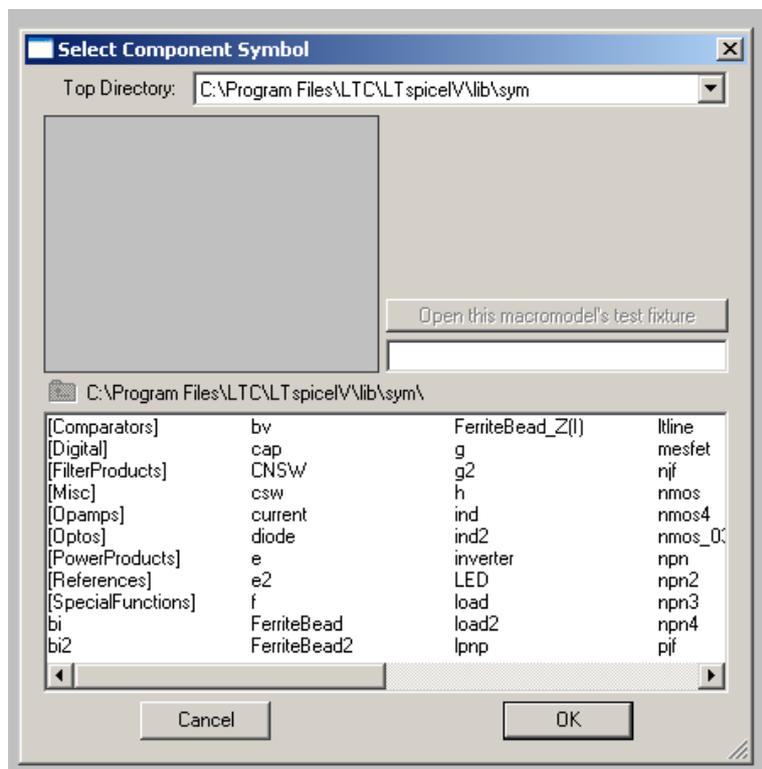


2. The LTspice window will be as shown start with a new schematic by clicking on file new schematic or by clicking on the new schematic icon as shown below

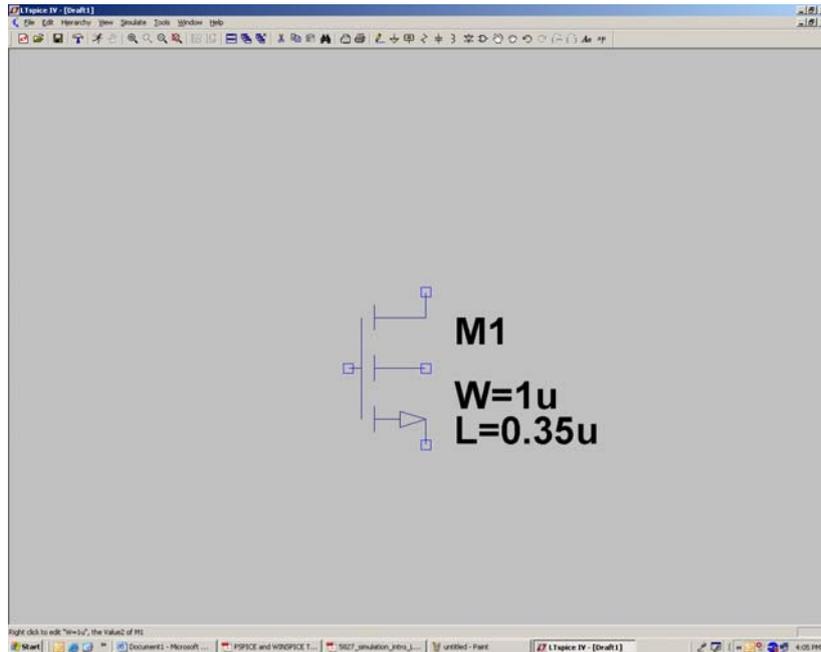


3. An empty schematic window is created .Click on file ->save as and save the schematic with the name “inverter”

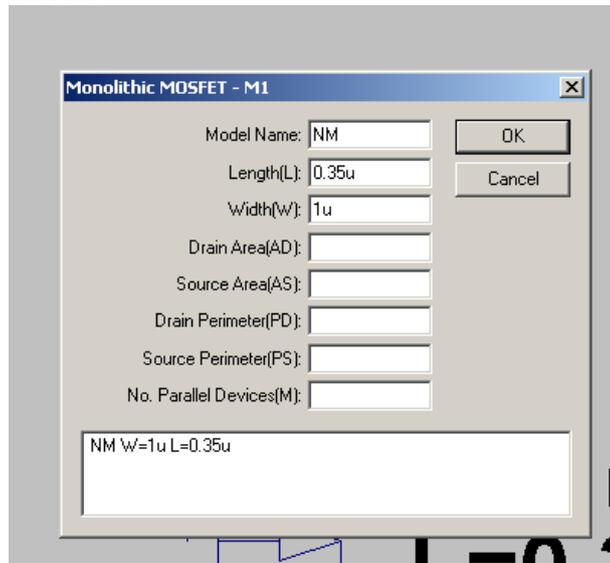
4. Click on **edit ->component** or press the **F2** key a popup window appears as shown below.



This window has the list of components available for your use. You can notice that the current and voltage sources are just represented as current or voltage. The type of source you need can be specified after adding the component to the schematic. To start our inverter design click on nmos\_035. And place it on the schematic window.



5. to change the Width or length right click on the component a popup window appears as shown below



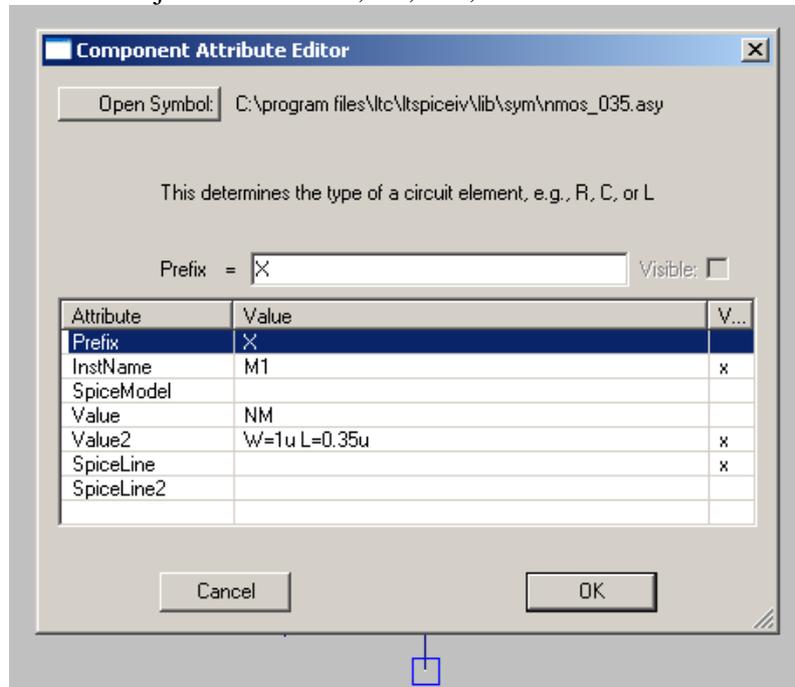
7. You can enter the Width and length in the pace provided. You can also enter the AD AS PD PS manually or you can configure it to be calculated automatically.

To enter manually you can calculate the values using the formula

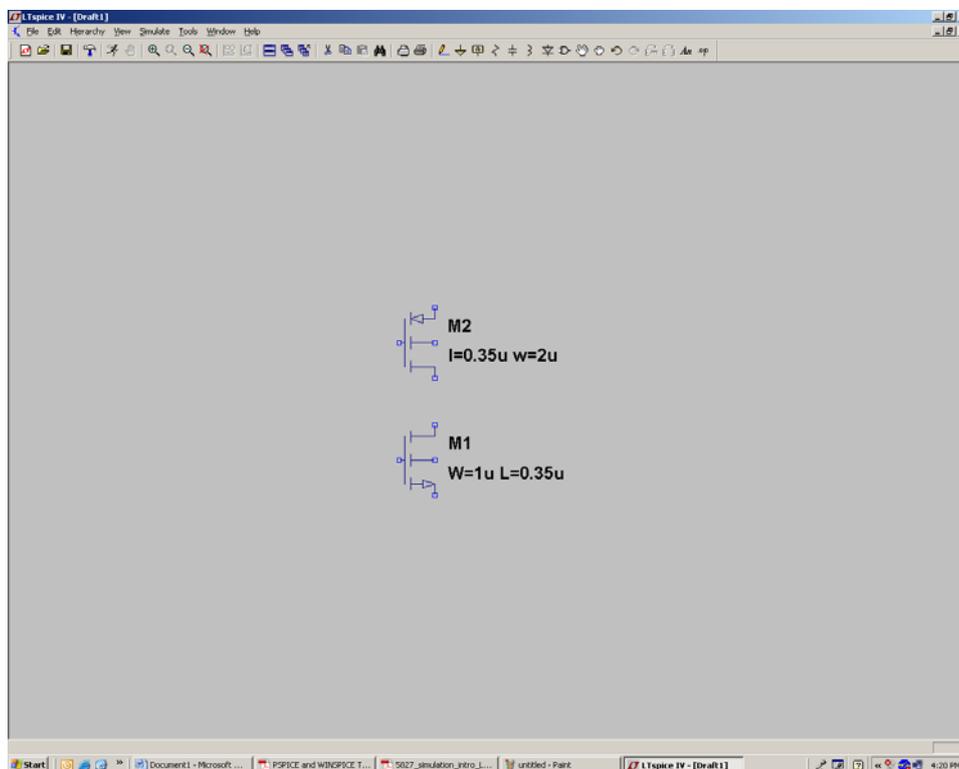
$$AD=AS=0.8p*W$$

$$PD=PS=4.0u+W$$

8. To configure the values automatically press and hold CTRL+Right click on the component a window will appear as shown below. Change **Prefix to X** to use sub circuit model with automatic adjustments of AS, PS, AD, PD



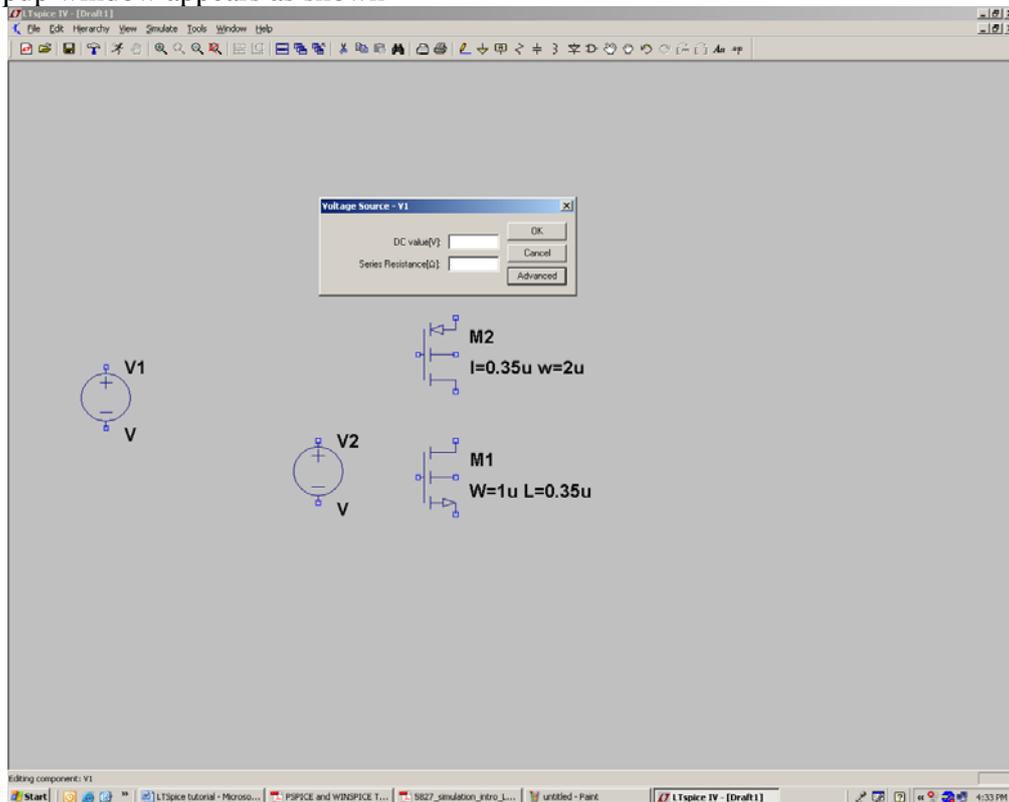
9. Repeat the same procedure for a PMOS transistor. The part name for the PMOS transistor is pmos\_035. The screen looks as shown below.



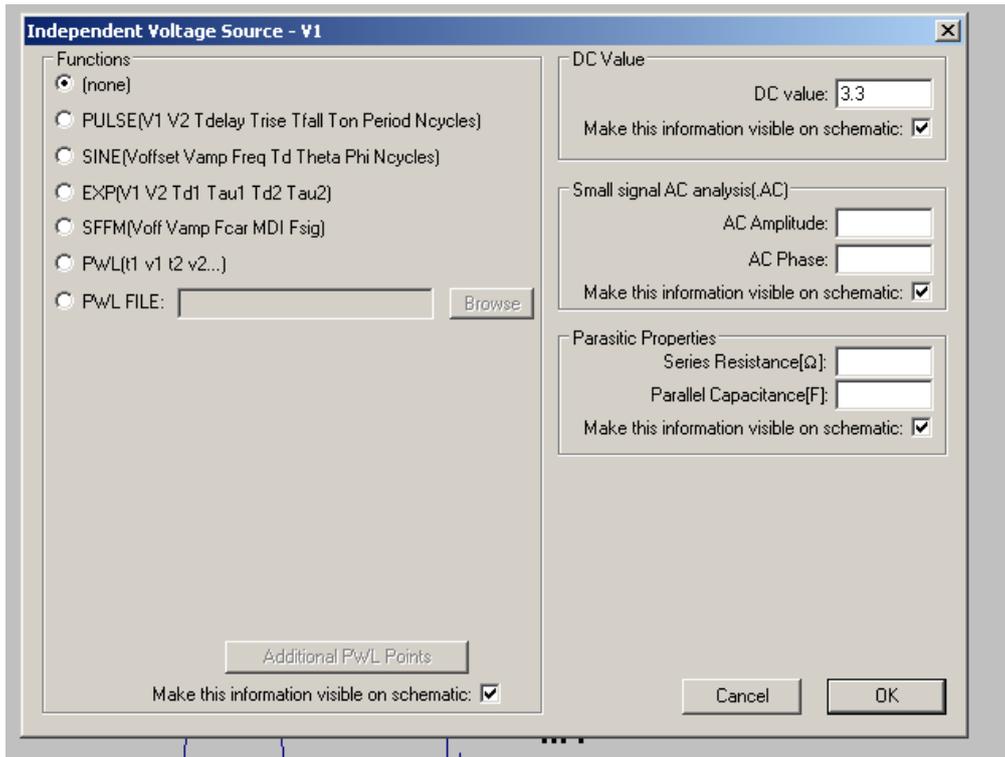
10. Now to place the ground press 'G' or click on edit->place ground.

11. To add a voltage source click on F2 key to get the "select component symbol window" and choose voltage and add the voltage source to the schematic. Add two instances of the voltage source one will be a DC source to give the supply to the inverter and the other will be a pulse source used as input to the inverter.

12. To configure the voltage sources V1 as a DC source right click on the component and a popup window appears as shown



Click on the advanced button to get the window as shown below.



This window provides you options to configure you voltage source as a dc source, ac source, pulse sine exponential etc. to configure V1 as a dc source just enter the dc value of 3.3 volts and click on ok.

**13.** To configure the voltage source v2 as pulse source right click on it and click on advanced button and then select the pulse radio button and configure the voltage source as shown below representing the various parameters of the pulse signal and click on ok.

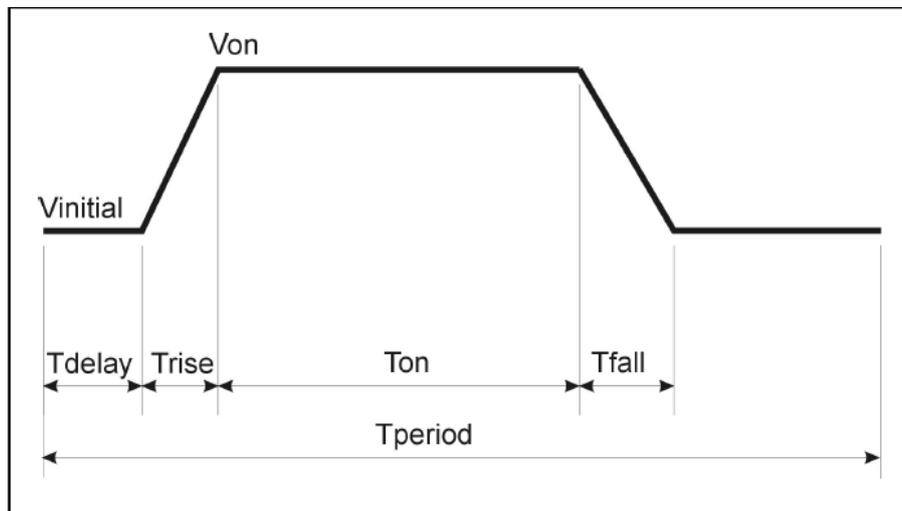
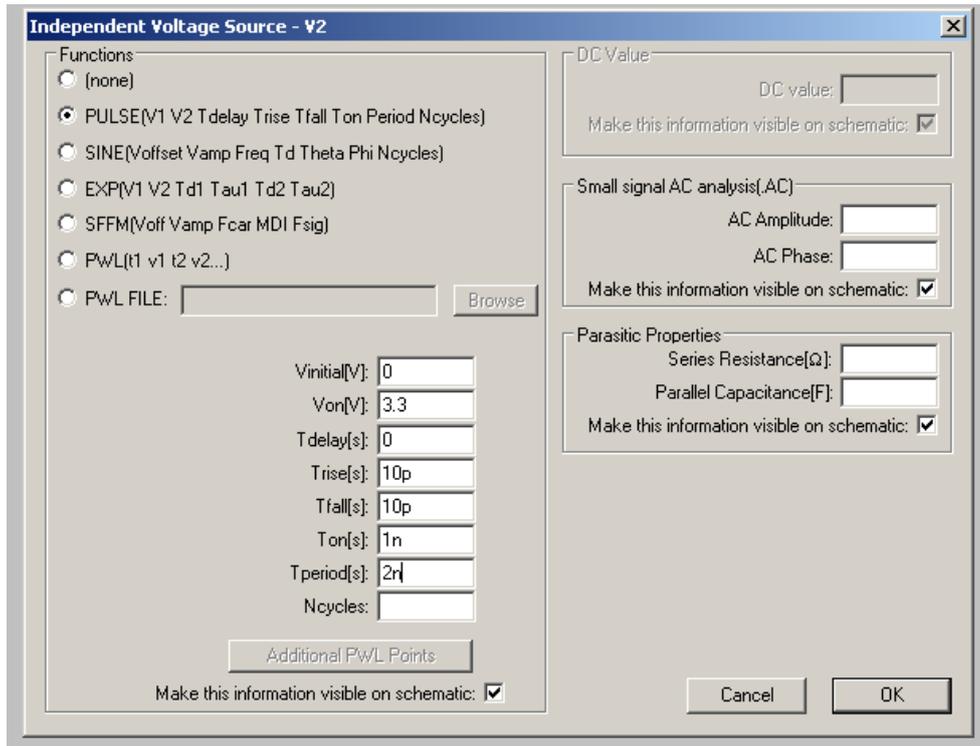
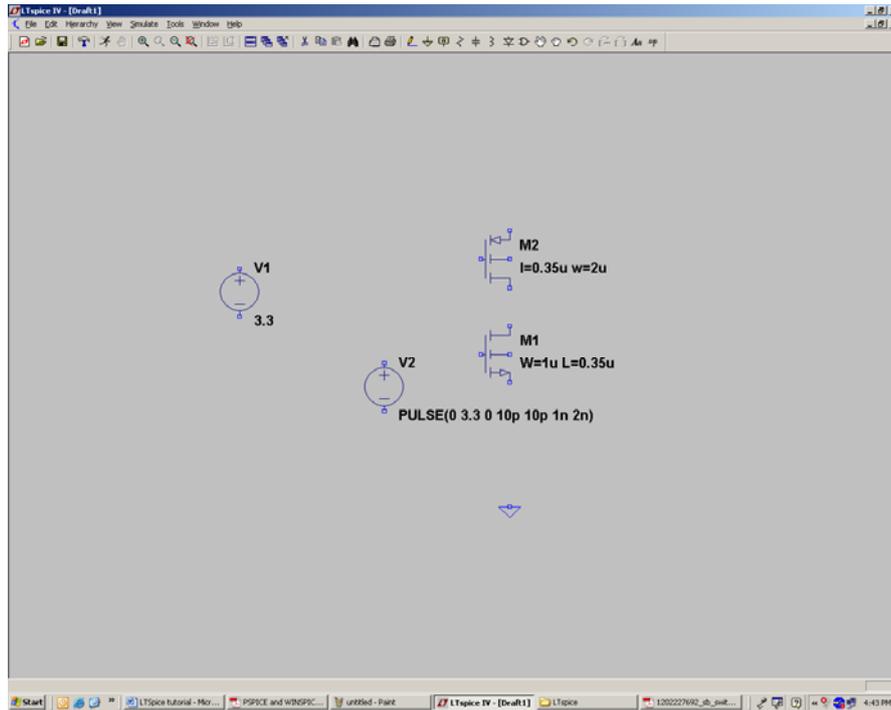
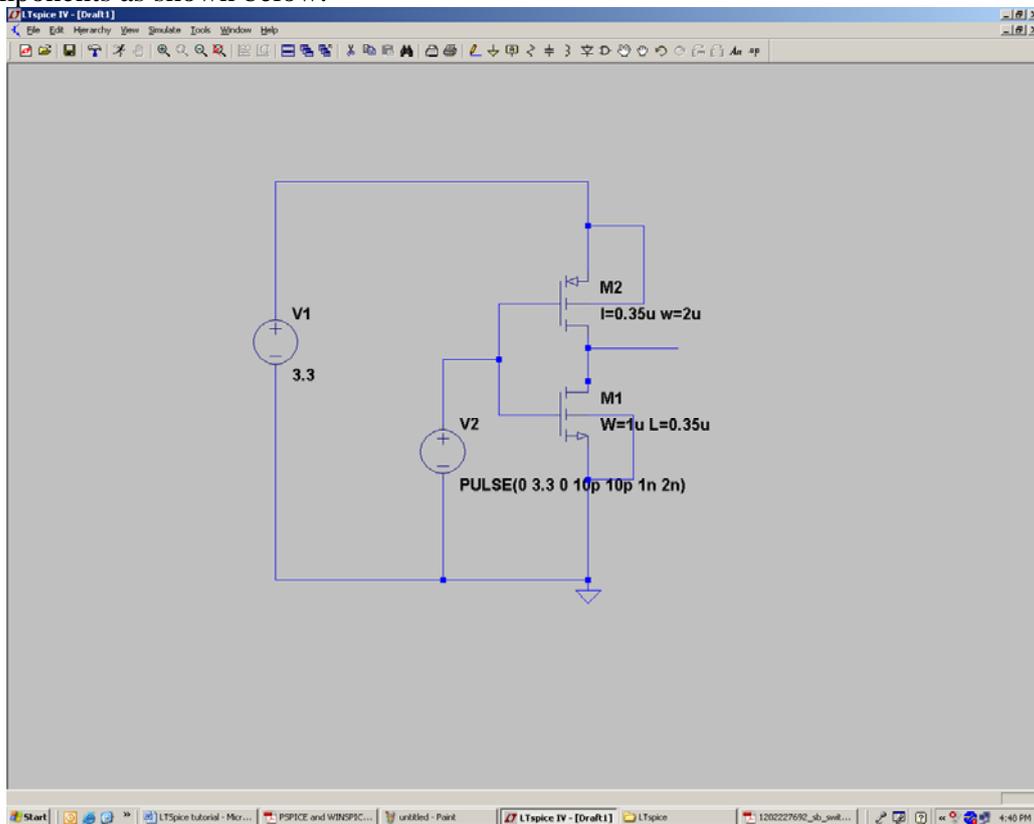


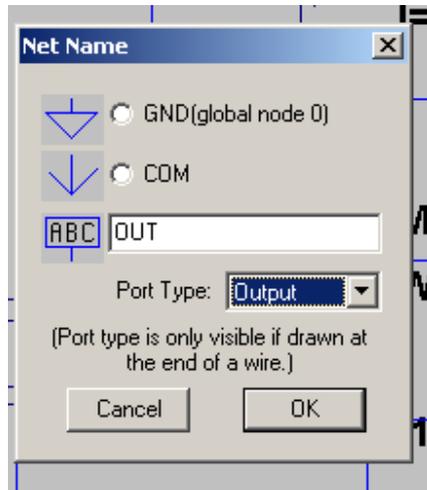
Figure taken from [1] shows how the parameters are related to the pulse signal.



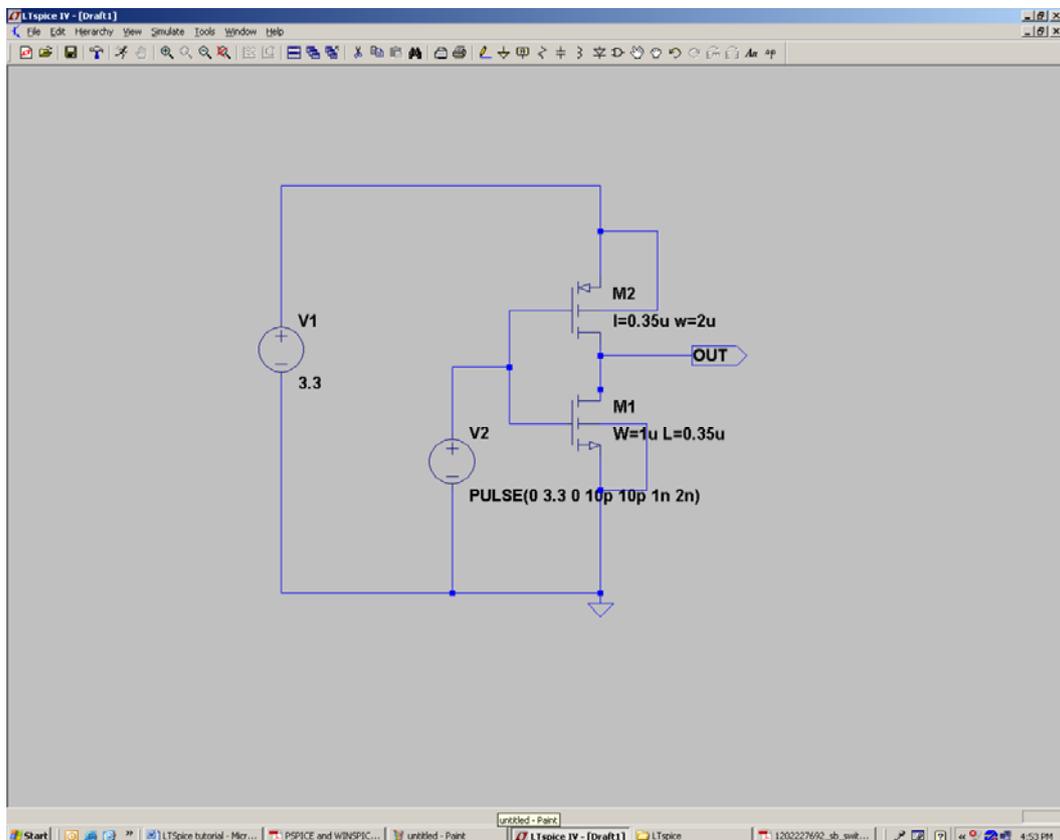
14. To connect the components use edit -> Draw wire or press F3 key and connect the components as shown below.



15. To add an output pin click on edit->label net or press F4 button a window as shown below appears

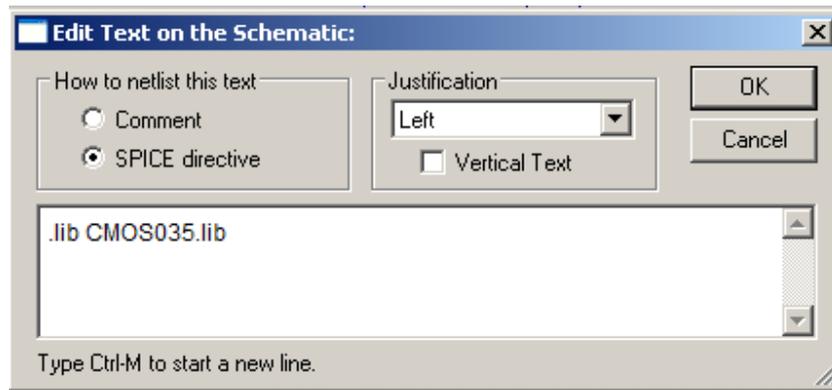


In this type the port name to be out and select the port type to be output and click on OK and place it at the end of the output wire(remember the port will appear only if you place it at the end of a wire else it will be takes as the net name). The schematic looks as follows.



**16 It is very important that before simulating the circuit you always need to include the library file CMOS035.lib in your schematic.** This library file has the parameters for the nmos and pmos models we have used in the design. To include this library click on

the  button or click on edit->Spice directive from the menu or press 'S' to get the window



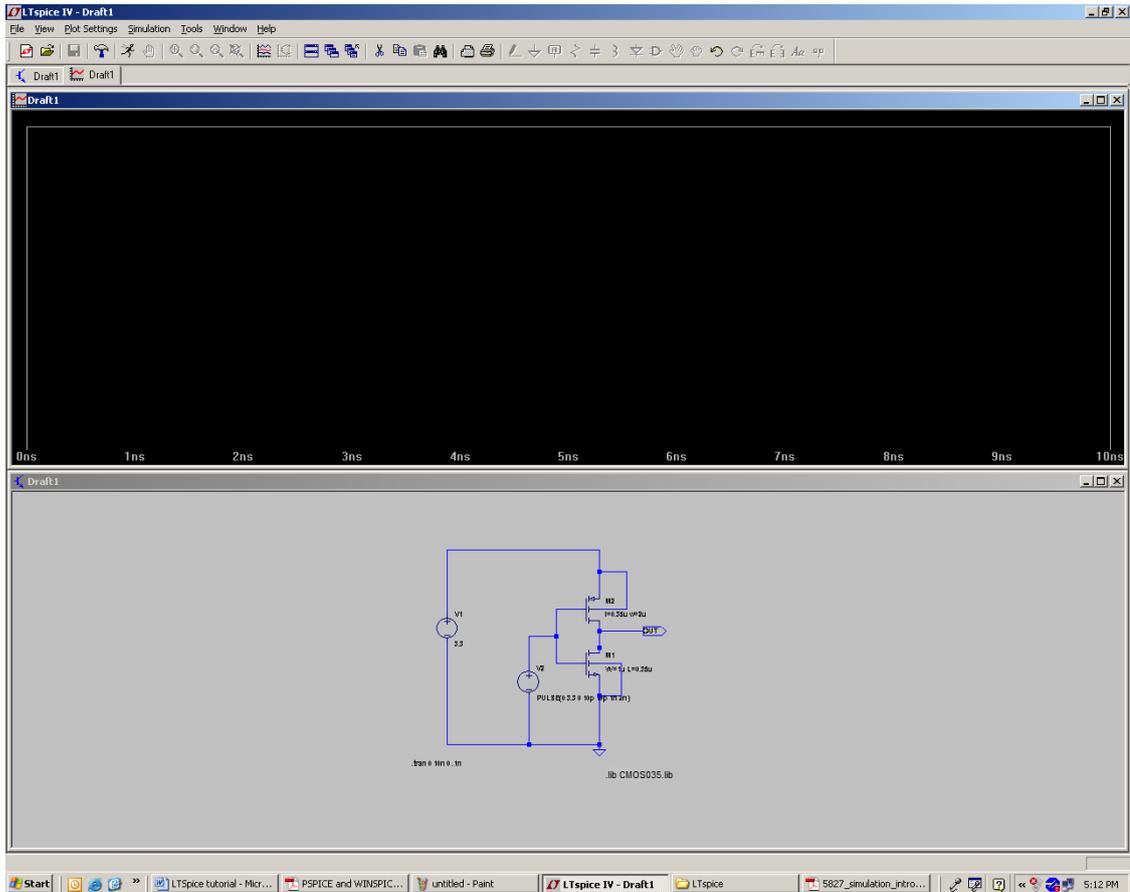
In the window enter the spice directive

`.lib CMOS035.lib`

To include the 0.35u mosfet transistor models for the transistors we have used in our design and click on ok and place it in the schematic.

### **Transient Analysis:**

1. From the simulation menu choose **Simulate->Edit Simulation Cmd.**
2. Click on the transient tab and in the *stop time* enter 10n in the *time to start saving data* enter 0 and *Maximum Time step* enter .1n and place it on the schematic.
3. This will make the circuit to run a transient simulation for 10ns with a time step of .1ns.
4. Click on simulate->run. Or the  button. A plot window will now popup as shown below.



To plot the input and output voltage signals of the inverter do as follows [2]

- Left click on any wire to plot the voltage on the waveform viewer

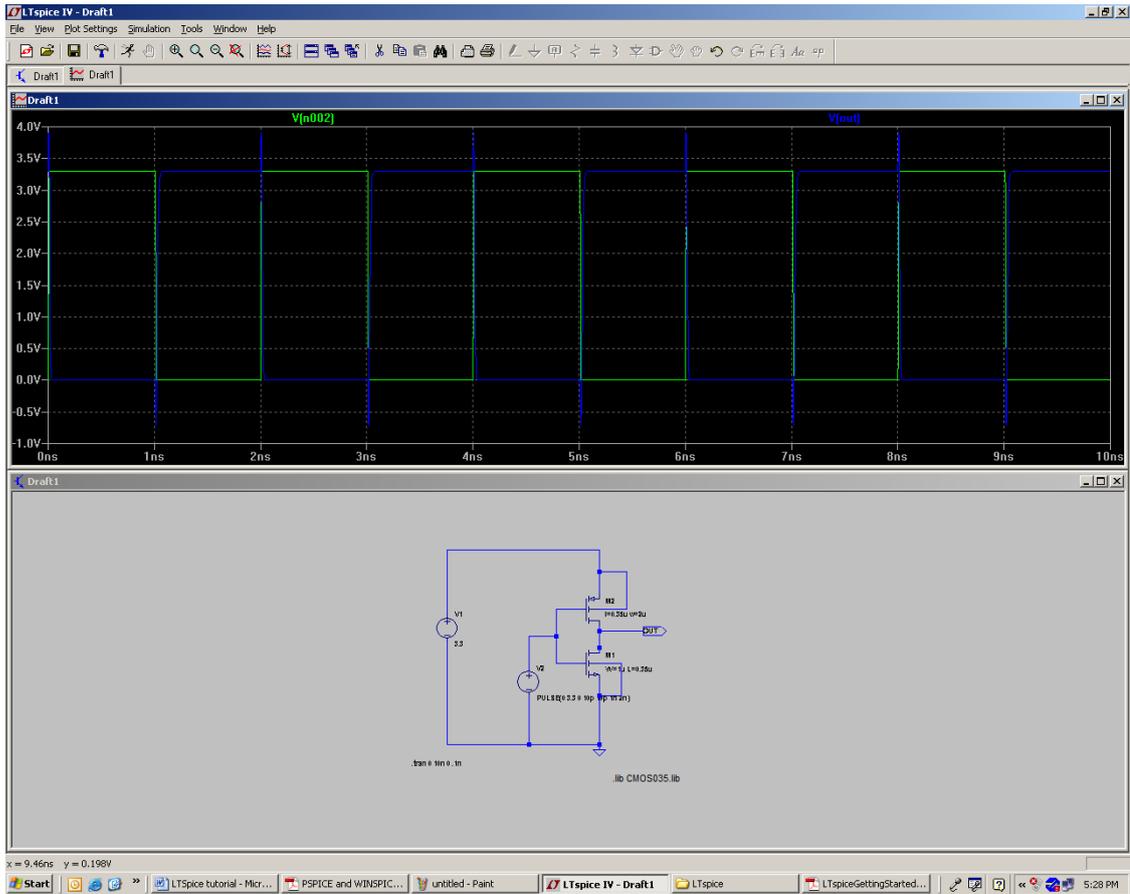


- Left click on the body of the component to plot the current on the waveform viewer(Convention of positive current is in the direction into the



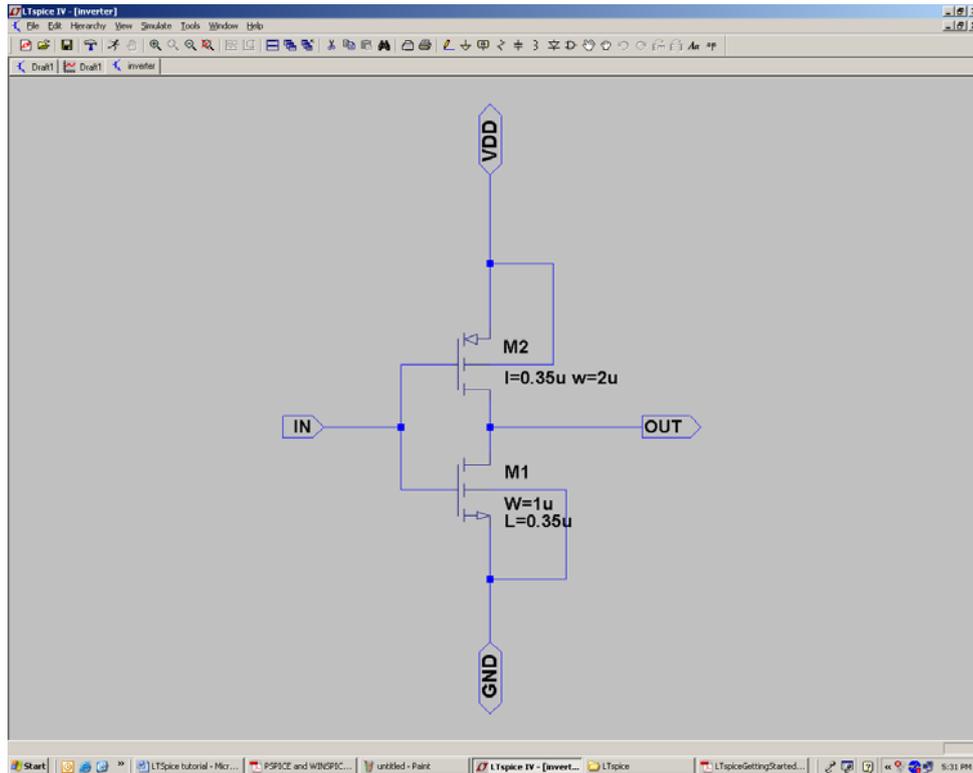
Or you can right click on the plot window and select add trace and from the list of voltages and currents listed you can select the signals you want to plot.

In the current example the signals plotted are the input and output voltages of the inverter. Left clicking on the inverter input net and the output net plots the signals as shown below.



## Creating a hierarchical design:

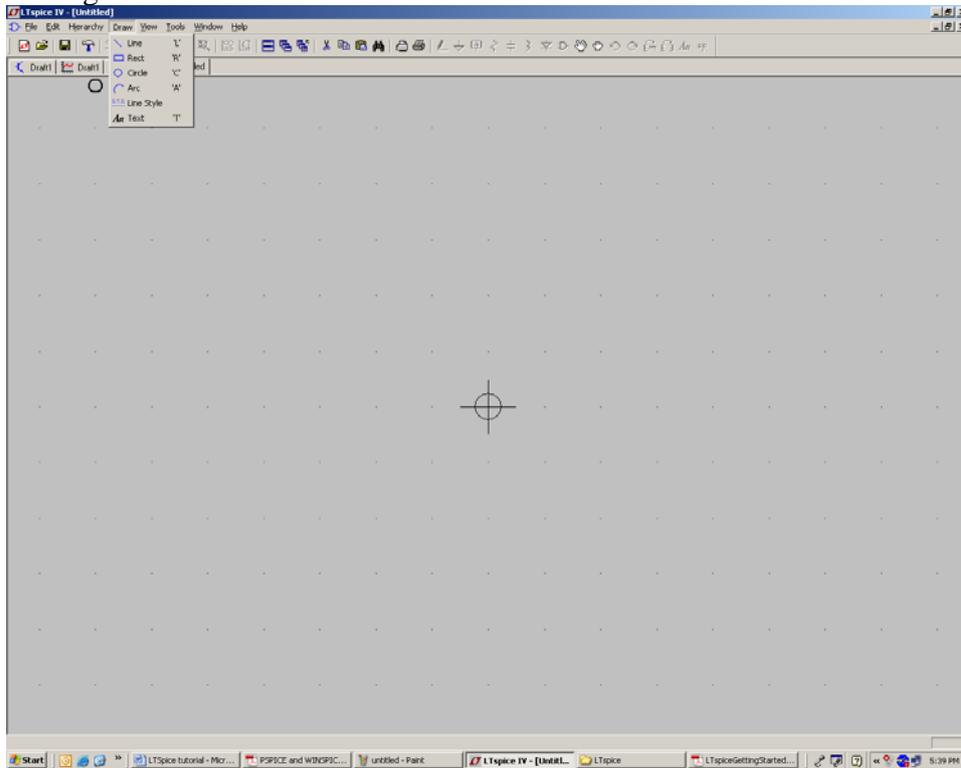
1. To create and hierarchical design create just the inverter circuit as shown below



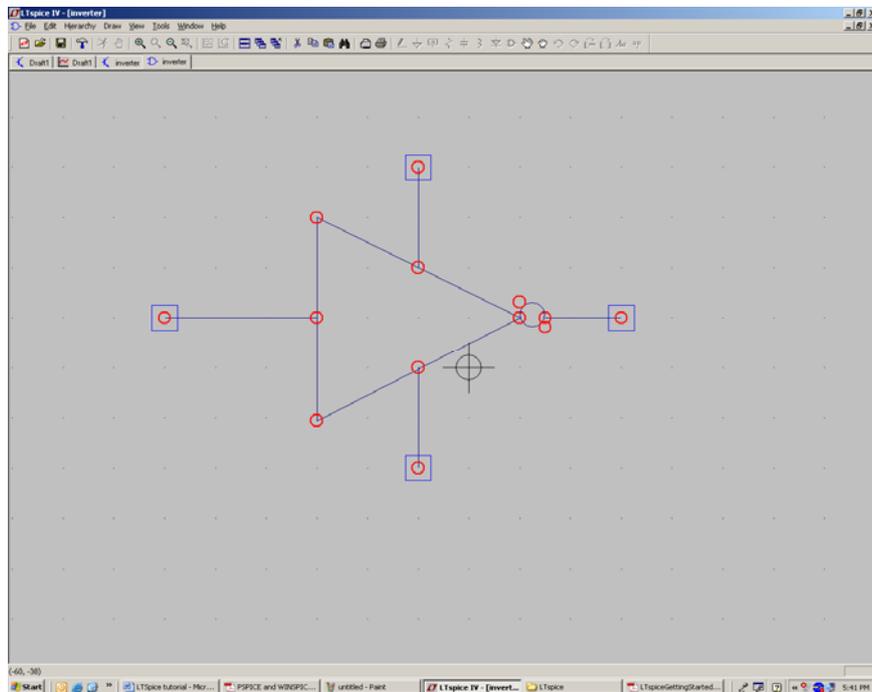
2. Here the pin out is created as before. Similarly press F4 and create a pin with name IN and select the port type as input and place it on the tip of the input wire .create pins called VDD and GND and select their port type as bi-direct place the pins on the tip of the respective wires and the circuit as shown above will be obtained. Save the schematic in the “C:\Program Files\LTC\LTspiceIV\lib\sym” directory with the name inverter.

3. Now to create a symbol for the inverter click on file-> new symbol and in the new window create the symbol for the circuit using the draw menu. You can draw lines

rectangles circles etc.

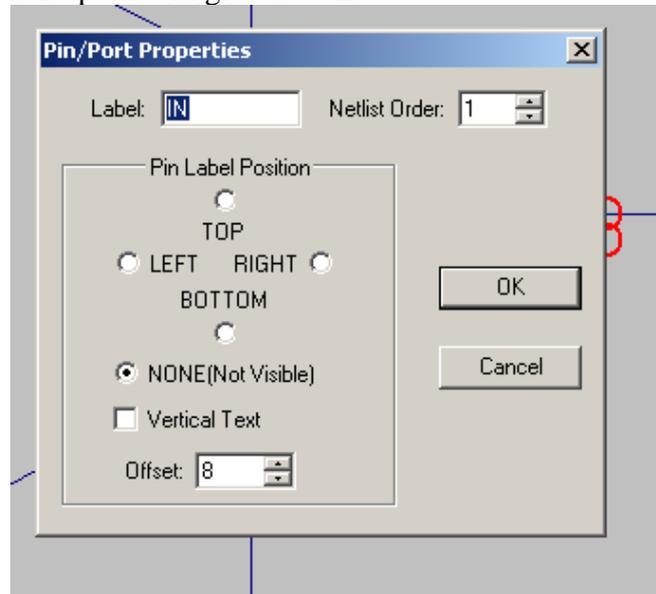


The symbol for the inverter we created so far is drawn and the output is as shown below



4. The ports are obtained by using edit->add port or by clicking the “P” button. The four rectangles the tip of each wire correspond to the four ports of the inverter we created in the schematic.

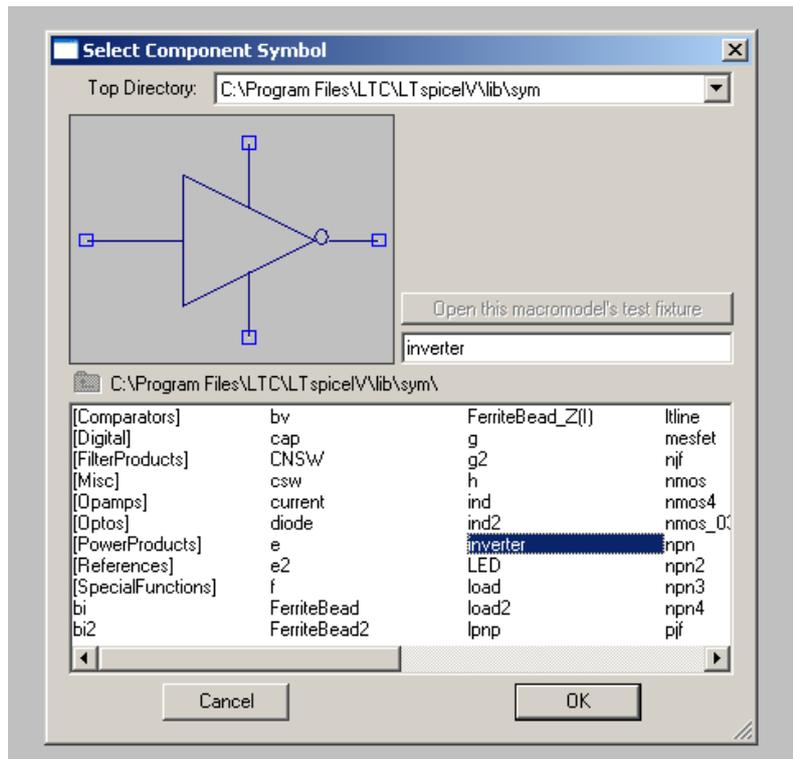
5. Right clicking on the ports will give the window



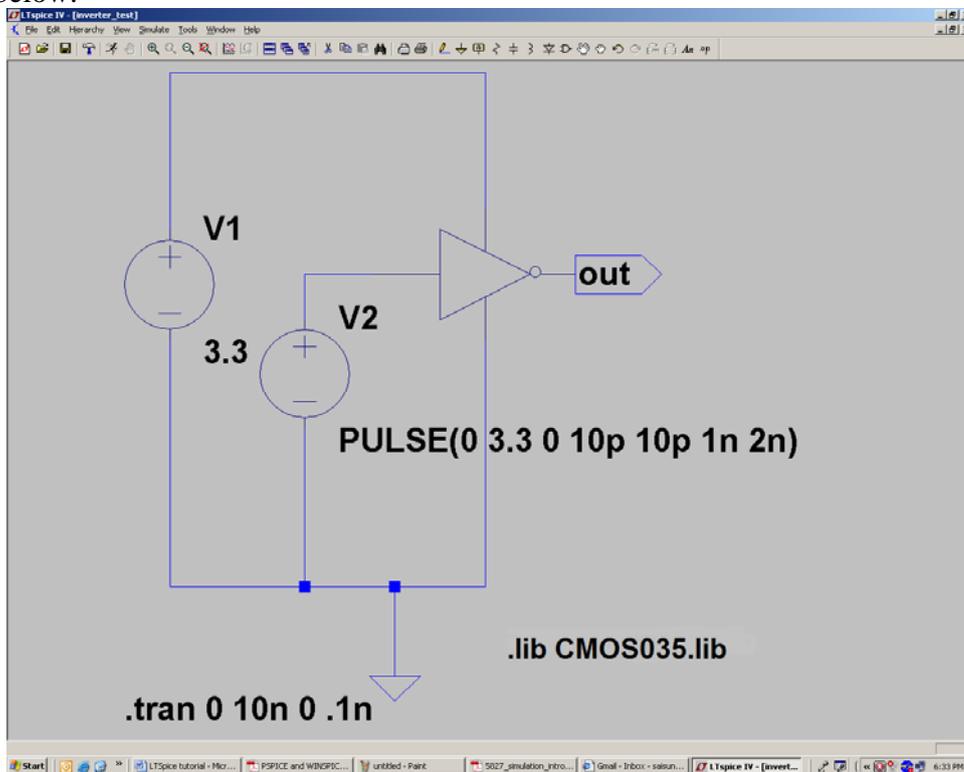
Using this window you can name the give name to each of the ports. Remember these names should be same as the names we gave in schematic.

6. Once the symbol is created save it under the directory “C:\Program Files\LTC\LTspiceIV\lib\sym”. The name of the symbol should be same as the name of the schematic. In our example it is inverter.

7. Close and open LT spice and create a new schematic inverter\_test and press F2 now in the component list you can find the inverter which you just created



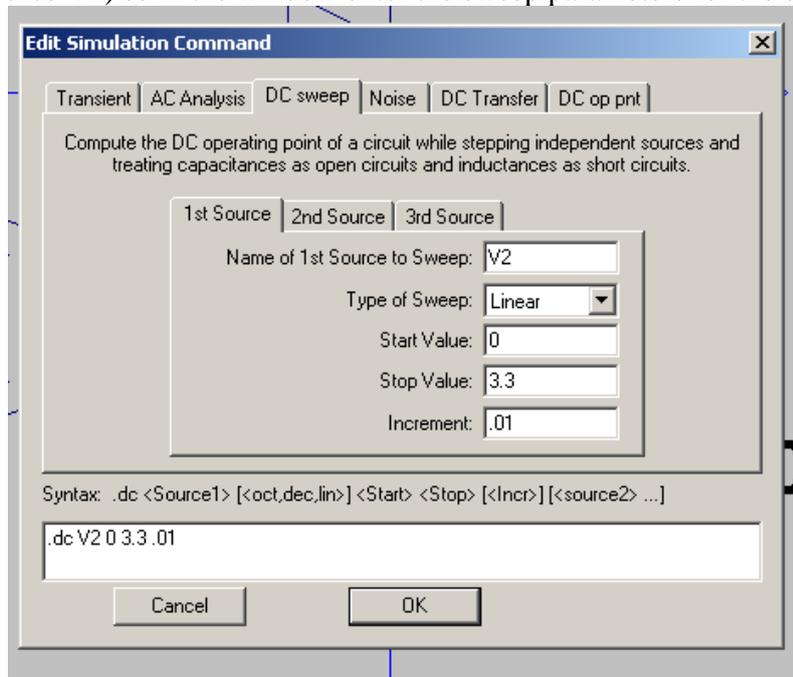
Add the inverter and add the voltage sources and ground as before and obtain the circuit as shown below.



8.Repeat the step 16 and then run the transient simulation.

## DC analysis:

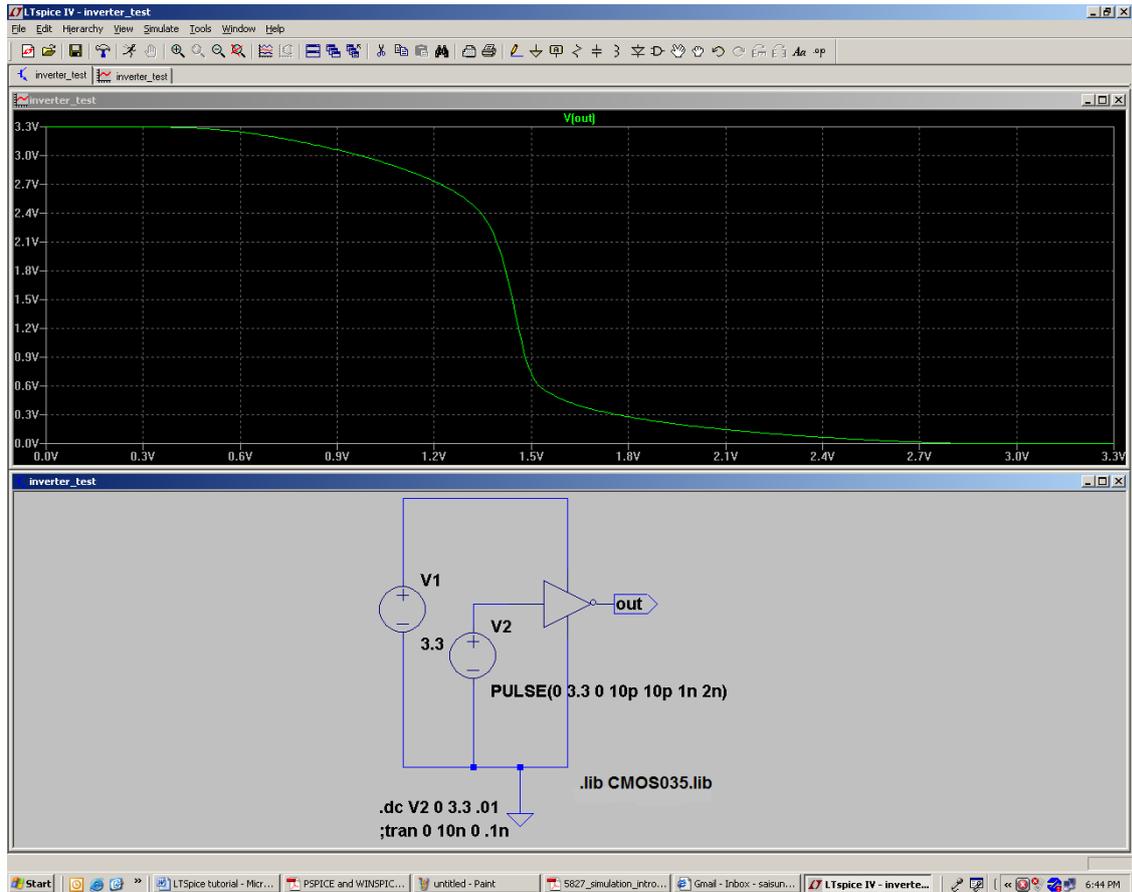
1. To do the DC analysis of the inverter click on **Simulate->Edit Simulation Cmd**
2. Select the DC sweep tab. To plot the output vs. input we need to sweep the input signal (i.e. voltage source V2) so in the window enter the sweep parameters for the simulation



The above screen shows that in the dc analysis the voltage source V2 will be swept from 0 to 3.3 volts with a step size of 0.01 volts. Click OK and place the command on the schematic.

3. Click on simulate->run. Or the  button. A plot window will now popup as shown below.

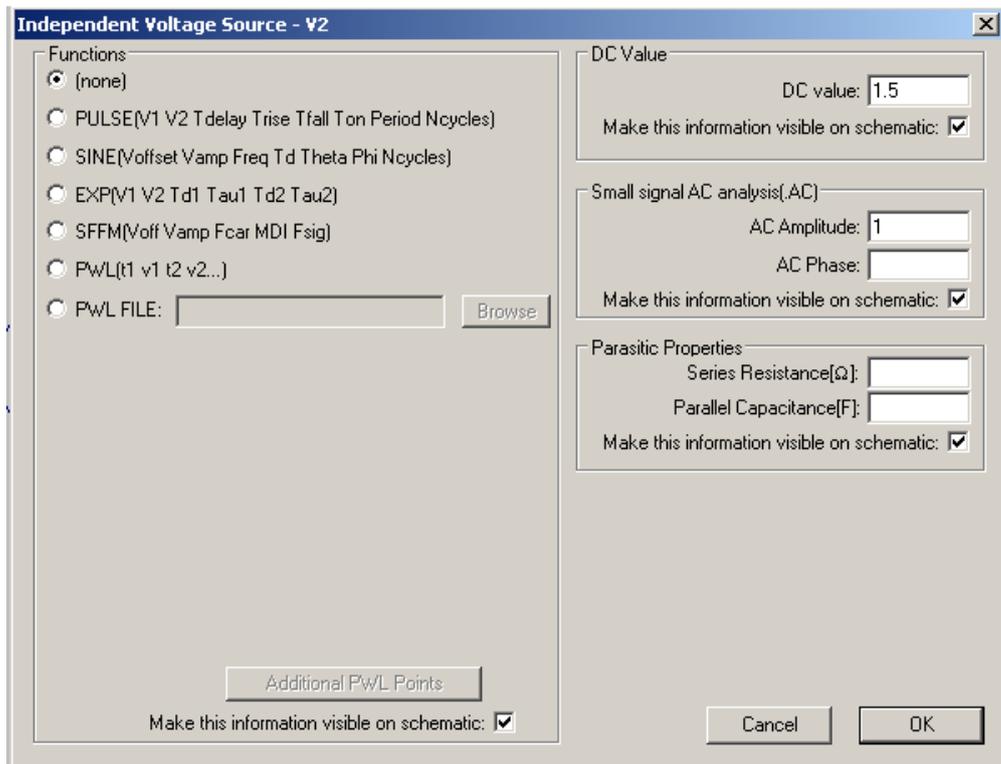
4. Probe the output voltage as before and the output looks as follows.



Note that the tab which is active when you close the Edit simulation command window decides which simulation will be run. The other simulation will be preceded by a semicolon on the schematic window and it will not run. In the previous case since the DC was active the dc simulation ran and the transient simulation got preceded by a semicolon n meaning it is not active now.

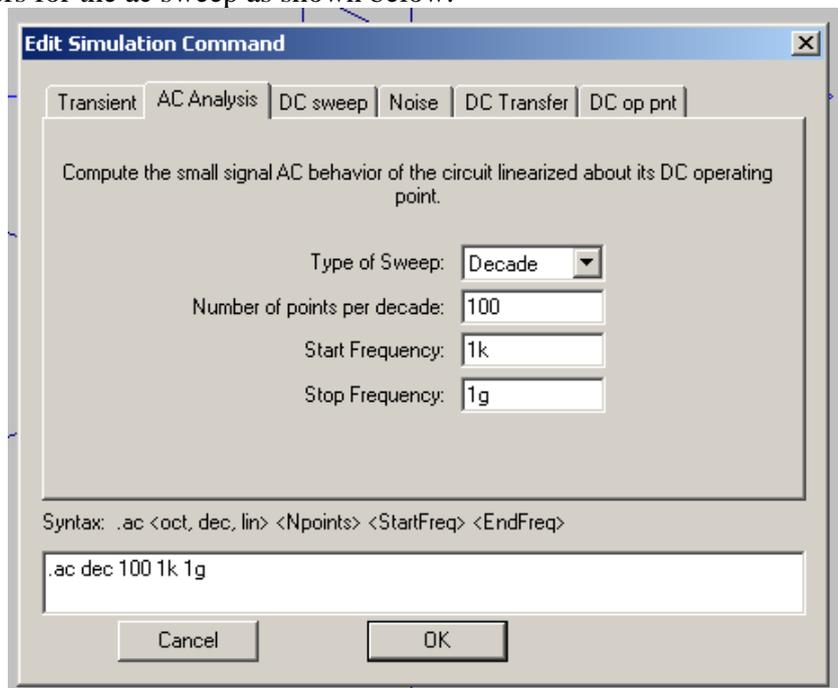
## Ac analysis:

1. To run the AC analysis we will use the inverter as an amplifier [3]. To do the analysis convert the input source V2 into an AC source.
2. to do this right click on the source V2 and select the none option



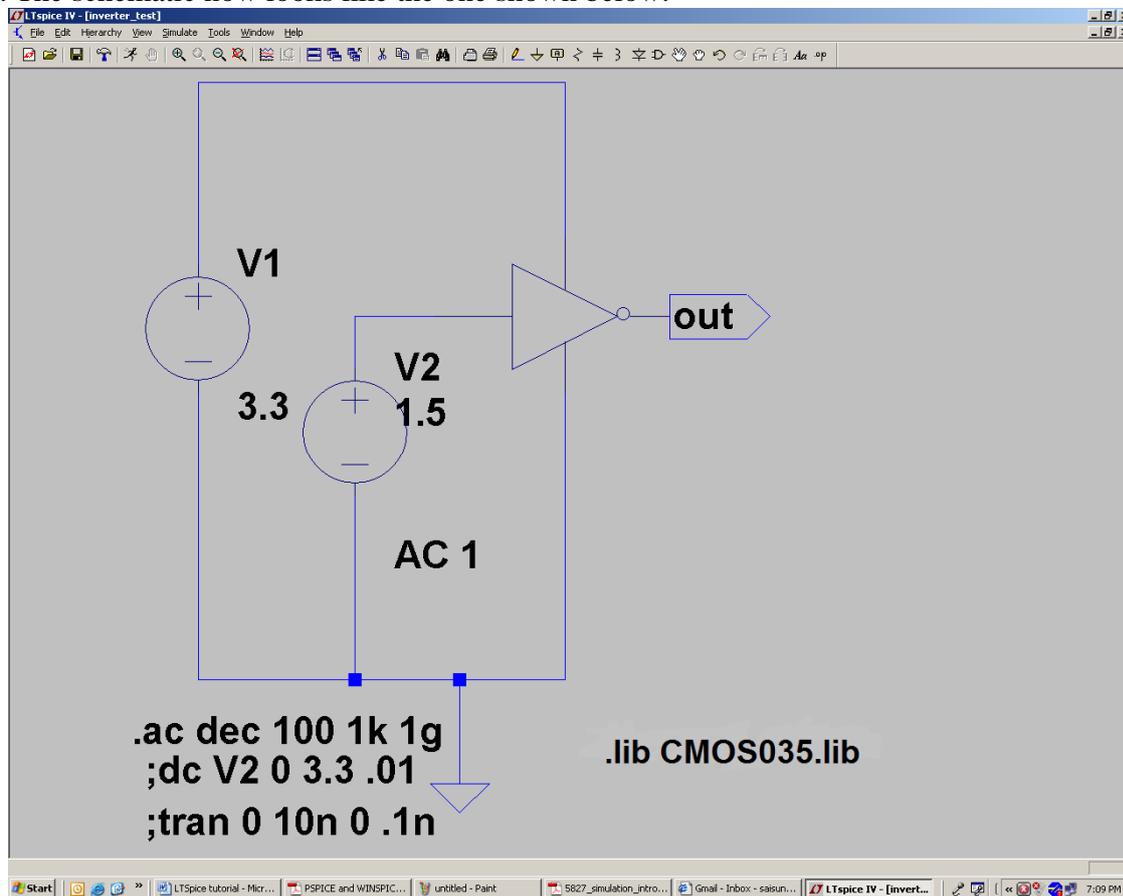
Enter a DC value of 1.5 volts and give an AC amplitude of 1 volts and click on ok.

3. Click on **Simulate->Edit Simulation Cmd.** Click on the AC analysis tab and then enter the parameters for the ac sweep as shown below.



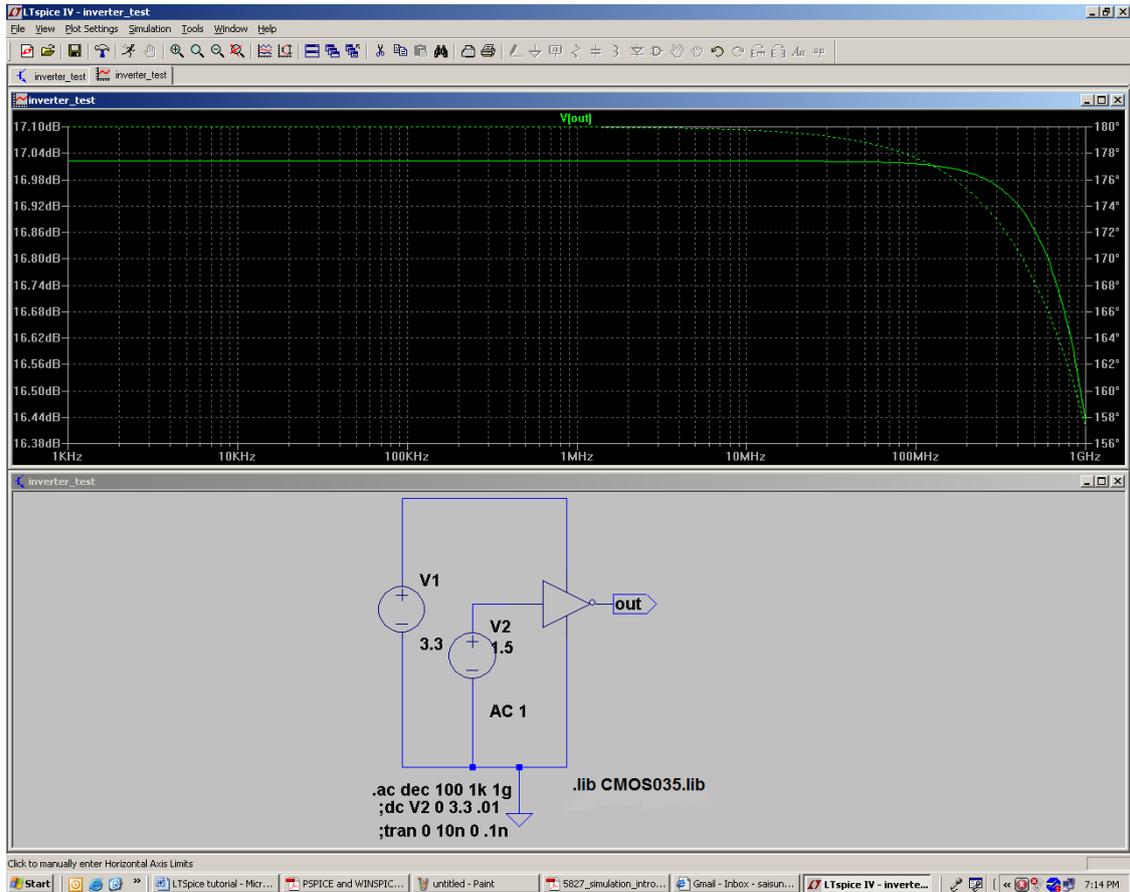
The above parameters indicate that the frequency sweep will in terms of decade. The number of points plotted per decade will be 100 the frequency is swept from 1 kHz to 1Ghz. click on ok and place the command in the schematic.

4. The schematic now looks like the one shown below.



5. Click on simulate->run. Or the  button. A plot window will now popup as shown below.

6. Probe the output voltage as before and you can see that the output amplitude in DB as well as the phase in degrees will be plotted in the graph.



The dotted line indicates the phase and the solid line indicates the gain in DB.

## References

- [1] "Introduction to Switcher CAD" from Linear technologies (<http://www.linear.com/>)
- [2] "LTspice Getting started guide" from Linear technologies (<http://www.linear.com/>)
- [3] "Introduction to schematic capture and Spice simulations using LTspice"

Web links for further reference:

1. <http://www.linear.com/>
2. <http://denethor.wlu.ca/ltspice/>
3. [www.elo.utfsm.cl/~elo106/materialLTspice/LTspice\\_guide.pdf](http://www.elo.utfsm.cl/~elo106/materialLTspice/LTspice_guide.pdf)